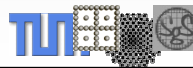


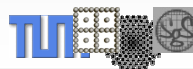
Case Study: CFD

- fluid mechanics as a prominent discipline of application for numerical simulations:
 - *experimental* fluid mechanics: wind tunnel studies, laser Doppler anemometry, hot wire techniques, ...
 - *theoretical* fluid mechanics: investigations concerning the derivation of turbulence models, e.g.
 - *computational* fluid mechanics (CFD): numerical simulations
- many fields of application:
 - aerodynamics: aircraft design, car design, ...
 - thermodynamics: heating, cooling, ...
 - process engineering: combustion
 - material science: crystal growth
 - astrophysics: accretion disks



Some Small Part of the World ...

- fluids and flows:
 - *ideal* or real fluids
 - ideal: no resistance to tangential forces
 - *compressible* or *incompressible* fluids
 - think of pressing gases and liquids
 - *viscous* or *inviscid* fluids
 - think of the different characteristics of honey and water
 - *Newtonian* and *non-Newtonian* fluids
 - the latter may show some elastic behaviour (e.g. in liquids with particles like blood)
 - *laminar* or *turbulent* flows
 - turbulence: unsteady, 3D, high vorticity, vortices of different scales, high transport of energy between scales
- typically: all require different models



The Mathematical Model

➤ starting point: continuum mechanics

- basic conservation laws (remember heat conduction in the modelling section): conservation of *mass* and *momentum*
- with the transport theorem and Newton's second law, we get

$$\frac{\partial}{\partial t} \rho + \operatorname{div}(\rho \vec{u}) = 0 \quad \text{continuity equation}$$

$$\frac{\partial}{\partial t}(\rho \vec{u}) + (\vec{u} \cdot \operatorname{grad})(\rho \vec{u}) + (\rho \vec{u}) \operatorname{div} \vec{u} - \rho \vec{g} - \operatorname{div} \sigma = 0 \quad \text{momentum equation}$$

- above quantities:

$\vec{u} = (u, v, w)$ velocity σ tension tensor

ρ density

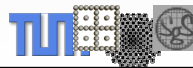
\vec{g} gravity

$$\operatorname{div} \vec{u} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

$$\operatorname{grad} p = \left(\frac{\partial p}{\partial x}, \frac{\partial p}{\partial y}, \frac{\partial p}{\partial z} \right)$$



Introduction to Scientific Computing
Lesson 14: Case Study: CFD



Slide 3

The Mathematical Model 2

➤ What to do with the tensor σ ?

- viscous case: not diagonal due to friction forces
- Newtonian case: isotrope, Stokes' postulate
- hence: pressure p and viscosity ν appear

➤ incompressible case: density is constant

➤ introducing Reynolds number Re (dimensionless, essentially reciprocal of viscosity and some scaling), we finally get the famous Navier-Stokes equations:

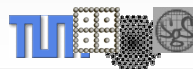
$$\frac{\partial}{\partial t} \vec{u} + (\vec{u} \cdot \operatorname{grad}) \vec{u} + \operatorname{grad} p = \frac{1}{Re} \Delta \vec{u} + \vec{g}$$

$$\operatorname{div} \vec{u} = 0$$

- two coupled PDE, nonlinear
- involving velocity and pressure, 1. and 2. spatial derivatives



Introduction to Scientific Computing
Lesson 14: Case Study: CFD

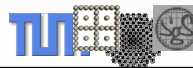


Slide 4

The Mathematical Model 3

➤ what about boundary conditions?

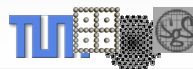
- **no-slip:**
 - The fluid can not penetrate the wall and sticks to it (tangential and normal velocity components are 0).
- **free-slip:**
 - The fluid can not penetrate the wall but does not stick to it (the normal velocity component and the tangential component's normal derivative are 0).
- **inflow:**
 - Both tangential and normal velocity components are prescribed (defined inflow).
- **outflow:**
 - Both velocity components do not change in normal direction (free outlet without restrictions).
- **periodic:**
 - Same velocity and pressure at inlet and outlet.



The Numerical Treatment

➤ discretization scheme: Finite Differences (can be shown to be equivalent to Finite Volumes, here)

- **grid:**
 - strictly orthogonal
 - staggered grid
- **spatial derivatives:**
 - Laplacian: standard 5- or 7-point stencil
 - first derivatives: mixture of central differences and upwind
 - derivatives of nonlinear terms: as first derivatives
- **time discretization:**
 - explicit Euler scheme (simple, but stability restrictions)
- **coupling of equations:**
 - Chorin's projection method; leads to a Poisson equation for the pressure
- **solution of SLE:**
 - SOR

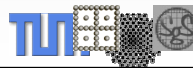


The Implementation

- geometry representation as a flag field (cf. *Marker-and-Cell*)
- input data (boundary conditions) and output data (computed results) as arrays
- modular C-code
- parallelization:
 - simple data parallelism, domain decomposition
 - straightforward MPI-based parallelization
- target architectures:
 - (real) parallel computers
 - clusters (NOW)
- The rest is (some ☺) programming!



Introduction to Scientific Computing
Lesson 14: Case Study: CFD



Slide 7

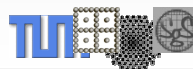
The Visualization

- techniques: all the stuff discussed before:
 - isosurfaces
 - orthoslices
 - streamlines
 - streaklines
 - particle tracing
 - ...
- finally some examples for visualized flows:

Ein paar NAST-Bildchen!



Introduction to Scientific Computing
Lesson 14: Case Study: CFD



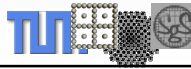
Slide 8

Outlook

- Studying this simulation cycle for CFD as presented very shortly here will be the topic of next semester's practical *Scientific Computing and Visualization*.
- There, you will develop your own simple simulation code and run your own fluid flow simulations, including some pretty pictures and movies to see what you've done.
- In addition to that, you will meet again many of the topics discussed in this introductory course during the CSE program – in more detail, and related to other topics or applications.
- For now, that's all 😊 - see you!



Introduction to Scientific Computing
Lesson 14: Case Study: CFD



Slide 9